

Numerical and Experimental Analysis of Transpiration Cooling in Carbon Dioxide
Atmosphere at Hypersonic Mach Numbers

V. N. Kulkarni, P. S. Kulkarni and K. P. J. Reddy

Department of Aerospace Engineering, Indian Institute of Science, Bangalore-560 012, India

Email: psk@aero.iisc.ernet.in

ABSTRACT

The numerical solutions are obtained for skin friction, heat transfer to the wall and growth of boundary layer along the flat plate by employing two dimensional Navier–Stokes equations governing the hypersonic flow coupled with species continuity equations. Flow fields have been computed along the flat plate in CO₂ atmosphere in the presence of transpiration cooling using air and carbon dioxide.

1. INTRODUCTION:

The main focus of hypersonics research in recent times is on the re-entry aerodynamics in order to develop protection system for the vehicles from severe thermal loads encountered during this phase. The aerodynamic heating is a major design concern for vehicles flying at hypersonic Mach numbers. The thermal load problem is handled by using an effective cooling system. The transpiration cooling in which a cold fluid is injected into the boundary layer through a porous media such that coolant issues forth as a continuous mass is an important cooling technique. Many experimental^[1] as well as analytical and numerical studies of transpiration cooling have been reported in the literature^[2,3]. Complete Navier-Stokes solutions in two-dimensional form for transpiration cooling in earth's atmosphere using air, helium and carbon dioxide as coolants have been reported recently using CFD techniques^[3-5]. The focus in recent times has been on the planet Mars and many probes have been sent to explore the life on Mars. This paper presents an analysis of transpiration cooling in carbon dioxide atmosphere by considering a simple flat plate geometry flying at hypersonic Mach number and the transpiration cooling is achieved by injecting air and carbon dioxide gases. The N-S equations governing the hypersonic flow coupled with species continuity equations are solved numerically using finite volume technique. The numerical results generated using this CFD code have been verified and also validated by conducting experiments in hypersonic shock tunnel of IISc.

2. GOVERNING EQUATIONS AND NUMERICAL ALGORITHM:

Assuming CO₂ as the test gas flowing at hypersonic Mach number while the transpiration cooling is achieved by injecting a coolant gas the basic Navier-Stokes (N-S) equations are modified with appropriate species continuity equations. The modified non-dimensional N-S equations in the conservative integral form are written as:

$$\frac{\partial \bar{U}}{\partial t} + \frac{1}{A} \oint_S \bar{F} \bar{n} dS = 0 \quad (1)$$

Where, $\bar{U} = \frac{1}{A} \iint U dA$ is the average value of the column vector of conserved variables

$U = [\rho, \rho u, \rho v, \rho E, \rho m_1]^T$ in the control volume and \bar{F} is the flux vector defined in Ref. [4]. Here, ρ is the density, u and v are the velocity components in the x and y directions, E is the total energy of the mixture, A and S are the area and boundary of the control volume, m_1 is the mass fraction of the species 1 and \bar{n} is the unit outward normal. The viscosity coefficient μ for the gas mixture is expressed as a function of temperature based on either Sutherland's law or Wilkes mixture rule[3].

Report Documentation Page				Form Approved OMB No. 0704-0188	
Public reporting burden for the collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to a penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.					
1. REPORT DATE 14 APR 2005		2. REPORT TYPE N/A		3. DATES COVERED -	
4. TITLE AND SUBTITLE Numerical and Experimental Analysis of Transpiration Cooling in Carbon Dioxide Atmosphere at Hypersonic Mach Numbers				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S)				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Department of Aerospace Engineering, Indian Institute of Science, Bangalore-560 012, India				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release, distribution unlimited					
13. SUPPLEMENTARY NOTES See also ADM001800, Asian Computational Fluid Dynamics Conference (5th) Held in Busan, Korea on October 27-30, 2003., The original document contains color images.					
14. ABSTRACT					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 3	19a. NAME OF RESPONSIBLE PERSON
a. REPORT unclassified	b. ABSTRACT unclassified	c. THIS PAGE unclassified			

The CFD code developed is based on the finite volume method. Discretising and applying to a cell (i, j) the above equations lead to:

$$\frac{\partial U_{ij}}{\partial t} + \frac{1}{A_{ij}} \left\{ \sum_{l=1}^4 \bar{F} \cdot \bar{n} ds \right\}_{ij} = 0 \quad (2)$$

In the present work, Roe's scheme is used to determine the inviscid fluxes. The quantities such as u , v , T , $\partial u / \partial x$, etc., are required on an interface in order to evaluate the viscous fluxes. In the evaluation of the derivatives like $\partial u / \partial x$ and $\partial u / \partial y$ at an interface the auxiliary cell approach is used. The resulting set of equations is treated with the Lind-Jacobi relaxation algorithm.

3. RESULTS AND DISCUSSIONS:

The code is used to compute flow fields at $M_\infty=6$ & 7 and $Re = 10^6$ and 5×10^6 . The variation of normalized skin friction coefficient along the flat plate for $M_\infty=6$ and $Re = 10^6$ for both the injecting gases for different blowing parameter is shown in Fig 1. The CO_2 is found to be more effective in reducing the skin friction. The variation of temperature in the normal direction and the variation in wall temperature for the same Mach number and Reynolds number are shown in Figs 2 & 3. The results clearly show that the thermal boundary layer gets affected significantly due to blowing. It is also clear that the injecting gas reduces the wall temperature significantly. The experimental investigations have also been carried out to validate the CFD results and will be presented in the full length paper.

REFERENCES

- [1] Libby PA, Cresci RJ. Experimental investigation of the downstream influence of stagnation point mass transfer. *Journal of Aeronautical Sciences*, 1961, 28: 51-64
- [2] Hartnett JP, Eckert ERG. Mass transfer cooling in a laminar boundary layer with constant fluid properties. *Trans. Amer. Society for Mechanical Engineers*, 1957, 79: 247
- [3] Sreekanth, Reddy NM. Transpiration cooling analysis at hypersonic Mach numbers using Navier-Stokes equations. *AIAA Paper 94-2075* (1994)
- [4] Ravi BR. Transpiration cooling analysis including binary diffusion using 2-D Navier-Stokes equations at hypersonic Mach numbers. MS dissertation, Dept. of Aero Engg., IISc, Bangalore, India (1997).
- [5] PS Kulkarni, Ravi BR and KPJ Reddy "Two dimensional Navier-Stokes Solutions for transpiration cooling at hypersonic Mach numbers" 9th ACFM, IRAN

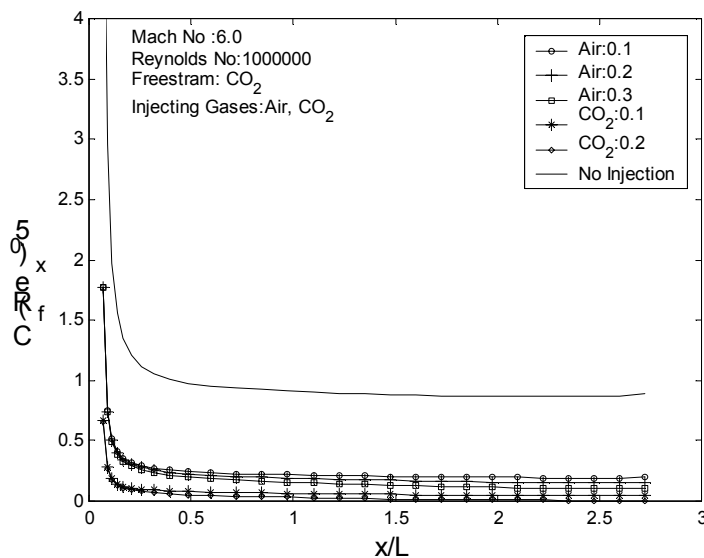


Fig 1. variation of skin friction for injection of different gases for $M = 6.0$ and $R = 10^6$

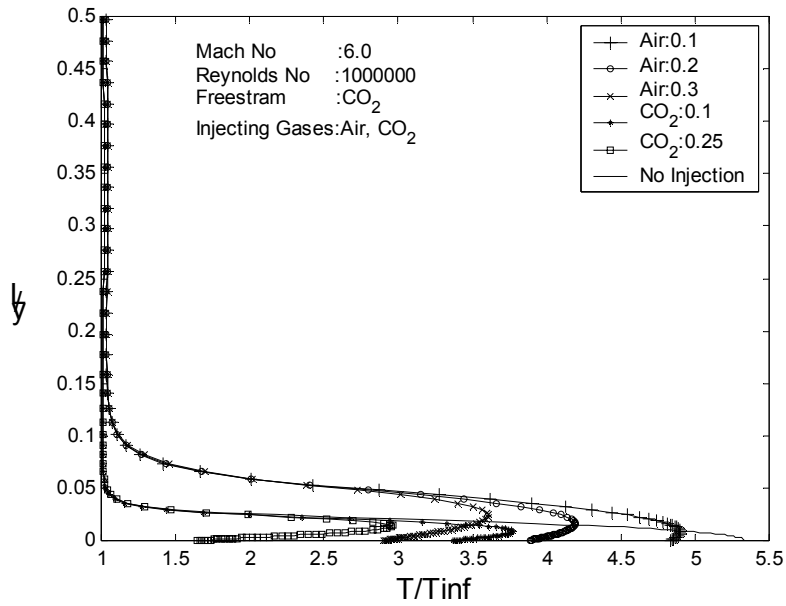


Fig2. variation of temperature profile in normal direction for $M = 6.0$ and $R = 10^6$

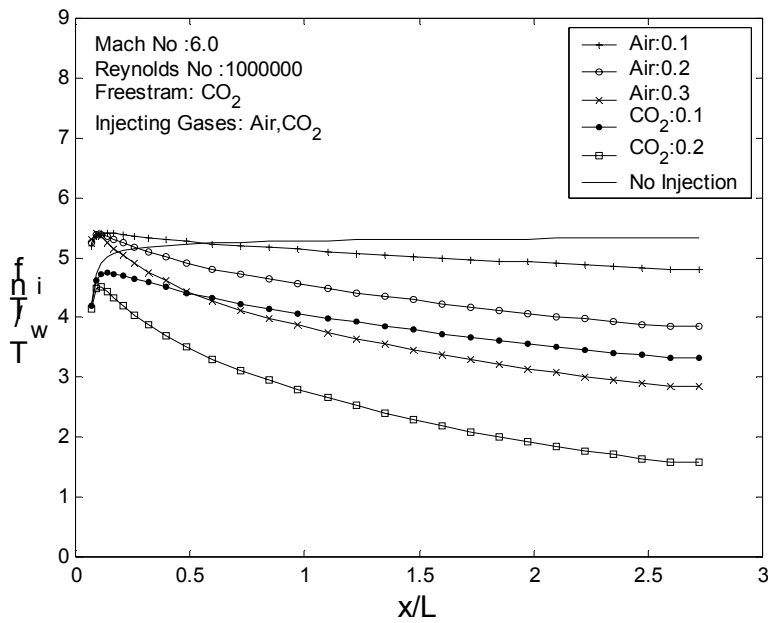


Fig3. Variation in wall temperature for different injecting gases for $M = 6.0$ and $R = 10^6$